Lecture Notes in Mechanical Engineering

Arun K. Saha
Debopam Das
Rajesh Srivastava
P.K. Panigrahi
K. Muralidhar *Editors*

Fluid Mechanics and Fluid Power – Contemporary Research

Proceedings of the 5th International and 41st National Conference on FMFP 2014

Springer

Lecture Notes in Mechanical Engineering

About this Series

Lecture Notes in Mechanical Engineering (LNME) publishes the latest developments in Mechanical Engineering—quickly, informally and with high quality. Original research reported in proceedings and post-proceedings represents the core of LNME. Also considered for publication are monographs, contributed volumes and lecture notes of exceptionally high quality and interest. Volumes published in LNME embrace all aspects, subfields and new challenges of mechanical engineering. Topics in the series include:

- Engineering Design
- Machinery and Machine Elements
- Mechanical Structures and Stress Analysis
- Automotive Engineering
- Engine Technology
- Aerospace Technology and Astronautics
- Nanotechnology and Microengineering
- Control, Robotics, Mechatronics
- MEMS
- Theoretical and Applied Mechanics
- Dynamical Systems, Control
- Fluid Mechanics
- Engineering Thermodynamics, Heat and Mass Transfer
- Manufacturing
- Precision Engineering, Instrumentation, Measurement
- Materials Engineering
- Tribology and Surface Technology

More information about this series at http://www.springer.com/series/11236

Arun K. Saha · Debopam Das
Rajesh Srivastava · P.K. Panigrahi
K. Muralidhar
Editors

Fluid Mechanics and Fluid Power – Contemporary Research

Proceedings of the 5th International and 41st National Conference on FMFP 2014

Springer

Editors
Arun K. Saha
Department of Mechanical Engineering
Indian Institute of Technology Kanpur
Kanpur, Uttar Pradesh
India

P.K. Panigrahi
Department of Mechanical Engineering
Indian Institute of Technology Kanpur
Kanpur, Uttar Pradesh
India

Debopam Das
Department of Aerospace Engineering
Indian Institute of Technology Kanpur
Kanpur, Uttar Pradesh
India

K. Muralidhar
Department of Mechanical Engineering
Indian Institute of Technology Kanpur
Kanpur, Uttar Pradesh
India

Rajesh Srivastava
Department of Civil Engineering
Indian Institute of Technology Kanpur
Kanpur, Uttar Pradesh
India

ISSN 2195-4356 ISSN 2195-4364 (electronic)
Lecture Notes in Mechanical Engineering
ISBN 978-81-322-2741-0 ISBN 978-81-322-2743-4 (eBook)
DOI 10.1007/978-81-322-2743-4

Library of Congress Control Number: 2016931198

Printed on acid-free paper

This Springer imprint is published by SpringerNature
The registered company is Springer (India) Pvt. Ltd.

Preface

The 5th International and 41st National Conference on Fluid Mechanics and Fluid Power was held in the campus of Indian Institute of Technology Kanpur during December 12–14, 2014. A total of 450 scientific participants, 275 of whom were students, attended the conference. The participants had fruitful scientific and technical discussions and exchanges that made the conference a complete success.

The overall response to the conference was quite encouraging. A large numbers of papers were received. After careful review, 397 papers were chosen for the inclusion for presentation during the conference. Among these, 286 papers were chosen for oral presentation while the rest were displayed in the form of posters. Subsequent to conference, an additional round of review was carried out. Among the presented papers, 161 papers were selected for inclusion in the published proceedings.

The papers appearing in the proceedings are distributed into 14 major parts:

1. Fundamental Issues and Perspective in Fluid Mechanics
2. Measurement Techniques and Instrumentation
3. Computational Fluid Dynamics
4. Instability, Transition, and Turbulence
5. Turbomachinery
6. Multiphase Flows
7. Fluid-structure Interaction and Flow-induced Noise
8. Microfluidics
9. Bio-inspired Fluid Mechanics
10. IC Engines and Gas Turbines
11. Transport Phenomena in Materials Processing and Manufacturing
12. MHD and EHD Flows
13. Granular Flows
14. Nuclear Reactor Thermal Hydraulics

These papers represent the most recent research on the subject. The editors would like to thank all the authors and the anonymous referees for paying attention to the quality of the publications. Partial financial support for publishing the conference proceedings is due to the financial support received from IITK, ISRO, DST, DRDO, and DAE and companies, including Coolflo (Hyderabad), GE Global Research (Bangalore), TESSCORN (Bangalore), Laser Science Services (Mumbai), and TSI Instruments (Bangalore).

The contents of these proceedings reveal the breadth of current activities in different themes related to fluid mechanics. We hope they form a useful starting point for beginners as well as practitioners in this discipline.

Kanpur, India

Arun K. Saha
Debopam Das
Rajesh Srivastava
P.K. Panigrahi
K. Muralidhar

About the Conference

Over the years, the country has made substantial progress in the fields related to fluid mechanics and fluid power. This development can be attributed to rapid advancements in computational methods and experimental techniques. The outcome has immensely benefitted the industry in numerous ways. In order to disseminate research findings and academic contributions relevant to the industry, the National Society of Fluid Mechanics and Fluid Power (http://www.me.iitb.ac.in/~fmfp/) holds a national conference every year and international conference once in every 4 years, either at an academic institution or at a research organization.

The National Society of Fluid Mechanics and Fluid Power was established in 1973 at IIT Bombay as a professional society to promote activities in the fields of fluid mechanics and fluid power in India. The annual conference has been held (mostly) uninterrupted over the years and the one in December 2014 is the 5th International and 41st National in the series. The society provides a common platform for academicians, engineers, and technologists to come together and discuss progress made within the country and abroad on various aspects of fluid mechanics and fluid power.

A total of 750 papers were received in the International FMFP Conference, 2014 and after a proper review process, 397 papers were selected for technical presentation—both oral and poster. A plenary lecture and several keynote lectures were scheduled during the two and half days of the conference. Over 450 college faculty, scientists, engineers, and students participated in the conference in the cool and quite ambience of Indian Institute of Technology Kanpur (India).

We acknowledge the effort of various committees, dedicated staff and student volunteers who contributed towards the successful completion of the conference. The success of the conference is also due to the generous financial support received from organizations such as IITK, ISRO, DST, DRDO, and DAE and companies including Coolflo (Hyderabad), GE Global Research (Bangalore), TESSCORN (Bangalore), Laser Science Services (Mumbai), and TSI Instruments (Bangalore).

Contents

About the Editors

Dr. Arun K. Saha (Ph.D., 1999, IIT Kanpur) has experience in experiments and numerical simulation in fluid mechanics and heat transfer. The numerical and experimental work done by him has helped in improving the fundamental understanding of jets and wakes. Currently he works on active control of flow and heat transfer for drag reduction of vehicles and gas turbine respectively. He is an author of several research publications and chapters in textbooks. He has been at IIT Kanpur since November 2004.

Dr. Debopam Das (Ph.D., 1998, Indian Institute of Science Bangalore) works in the area of experimental and theoretical fluid mechanics. His research work encompasses both compressible flow and incompressible flow. Compressible flow research is primarily on flow and acoustic field associated with formation of compressible vortex ring and its interaction with shock, wall, or other generic shape objects. Incompressible flow research spans across different topics; transition of unsteady internal flow, free shear and buoyancy-driven flows, unsteady aerodynamics of birds' and insects' flights. He has also designed and fabricated several flapping wing flying MAV's of different sizes including a hovering MAV. In theoretical fluid dynamics, he has contributed towards an analytical solution method for unsteady internal flow when the pressure as well as the velocity is unknown. He is an author of several research publications. He has been at IIT Kanpur since December 2001.

Dr. Rajesh Srivastava has worked extensively on numerical simulation of flow and transport through variably saturated porous media. His work provides significant insight into the transport of reactive solutes in porous media. His research finds applications in remediation of contaminated sites as well as mining of minerals through heap leaching. His present work focuses on the effect of climate change on water resources and inverse techniques for characterization of porous media and identification of possible contaminant sources. He is an author of several research publications and two text books. He has been at IIT Kanpur since December 1997.

Dr. P.K. Panigrahi is presently working as Professor and Head in the Mechanical Engineering Department, IIT Kanpur. Previously, he was the Head of the Laser Technology Program. He obtained his MS and Ph.D. from LSU, Baton Rouge, USA. His research interests include development and implementation of optical techniques for characterization of transport processes in various devices/systems at macro-scale and micro-scale involving fluid flow, heat and mass transfer.

Dr. K. Muralidhar (Ph.D., 1985, University of Delaware) has conducted a wide spectrum of experiments and numerical simulation in fluid mechanics and heat transfer. He has contributed to the understanding of transport phenomena in porous media, wave dynamics, in addition to development of numerical algorithms and laser measurement of flow and thermal fields. His research finds applications in industrial processes such as enhanced oil recovery, methane hydrates and CO_2 sequestration, regenerators, and growth of optical crystals. His present work relates to interfacial transport, biomedical imaging, and inverse techniques. He is an author of several research publications, textbooks, and monographs. He has been at IIT Kanpur since July 1987.

Part I
Fundamental Issues and Perspective in Fluid Mechanics

Effect of Inlet Shear on the Flow Structures Associated with Elevated Jet in Cross-Flow at Low Reynolds Number

Sachidananda Behera and Arun K. Saha

Abstract A Direct numerical simulations (DNS) is carried out to study the highly three-dimensional flow field around a jet coming out of a stack and interacting with the cross-flow passing it. The major objective of the study is to assess the effect of shear strength of inlet velocity on the wake behavior and structures for elevated jets in cross-flow (EJICF). All the simulations are carried out for a Reynolds number of 250 based on the averaged inlet velocity and width of the stack and the blowing ratios considered are 0.5, 1.0 and 1.5. Effect of shear strength (0.025, 0.050 and 0.075) of inlet velocity is studied by solving the incompressible unsteady 3-D Navier-Stokes equations using a second order discretization for space and time. The solution algorithm is based on MAC method. The dynamics of flow associated with the flow field, like different wakes zones present in the flow are discussed with respect to the shear strength of free stream velocity. The vortical structure associated with EJICF is also studied.

Keywords DNS · Elevated jet in cross-flow · Vortical structures · Wake

1 Introduction

The flow field of a jet coming out of an opening normal to a cross-flow has received a large attention due to its widespread application. The jet may emit from a ground level source or from an elevated source (pipe, stack). A jet coming out from a ground level source are often encountered in practical applications, such as that of engineering and industrial interests like turbine blade cooling, fuel injection, gas turbine combustor and V/STOL aircraft. Where as an elevated jet discharging into a cross-flow arise in

Sachidananda Behera (✉) · A.K. Saha
Department of Mechanical Engineering, Indian Institute of Technology,
Kanpur 208016, India
e-mail: sbehera@iitk.ac.in

A.K. Saha
e-mail: aksaha@iitk.ac.in

A.K. Saha et al. (eds.), *Fluid Mechanics and Fluid Power – Contemporary Research*, Lecture Notes in Mechanical Engineering,
DOI 10.1007/978-81-322-2743-4_1

many situations of environmental fields, e.g., burner, stack flare, plume dispersion and discharge of effluents. The complexities associated with the flow field of elevated jet in cross-flow are due to interactions among the jet, jet-wake, stack-wake and boundary layer developed at the wall. Flow properties of jets emitted into cross-flows from an elevated source and that from a ground level source have been studied in details in literatures, largely the attention being on the ground level case.

From environmental point of view (plume dispersion and pollutant dispersion) one of the major parameter is the velocity ratio of jet-to-crosswind and Briggs [1] advocated that this ratio mostly known as blowing ratio should always be greater than 1.5 to avoid any downwash of pollutants which could affect the near environment of the stack. But Huang and Hsieh [2] were of the opinion that to avoid down- wash the blowing ratios should be equal or greater than 2.0 and they suggested this on the basis of their experimental work. Arora and Saha [3] through their numerical study showed how the flow physics of an EJICF is affected by the blowing ratio, also the effect of blowing ratio on the concentration field was studied.

An interaction between jets coming out of a stack source with that of a cross-flow has been a major area of interest for researchers in the past, and still provides a lot of scope for further research because of the complex flow structures associated with this kind of flow. The major flow structures associated with EJICF are jet shear layer vortices, different wake zones, horse shoe vortex and counter rotating vortex pair (CRVP). Two types of wakes are discussed in literatures in relation to EJICF, namely stack wake and jet wake [4]. As described by Moussa et al. [5] the jet wake is developed because of the shedding of vortices behind the jets, as the vertically ejecting jet itself behaves as a bluff body. The stack wake get developed because the stack behaves as an obstacle to the cross-flow and the vortex shedding pattern is very similar to that observed in case of bluff bodies [4].

Eiff et al. [4] advocated about the presence of two kinds of wakes in reference to EJICF and showed how vortices associated with the jet wake interact with wake associated with stack. To this point no work is reported on the effect of inlet velocity profile on the wakes associated with EJICF and therefore the present work intends to study the effect of shear strength of inlet free stream velocity on the flow structures.

2 Solution Methodology

The 3-D unsteady Navier-Stokes equations for incompressible flow were solved using finite difference method to study the interaction of a square jet and cross-flow. The governing equations used are all in non-dimensional form. To achieve the non-dimensional form of the governing equations, the velocities are non dimensionalised with respect to the mean velocity of cross-flow, u_m, and all length scales with respect to width (D) of the square stack. The pressure is non dimensionalised with respect to ρu_m^2 and time with D/u_m. The governing equations are solved on a non-uniform staggered grid arrangement such that the velocities are defined on

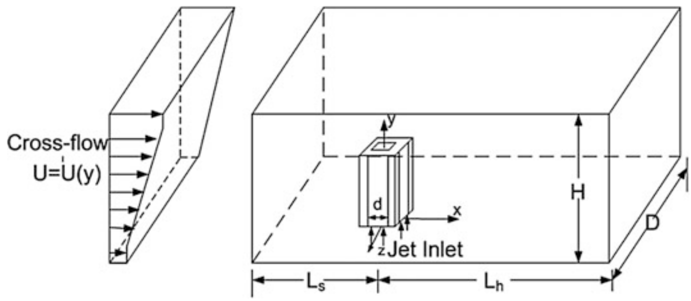

Fig. 1 Schematic of the computational domain

faces of the cell to which velocity is normal and pressure is defined at the center of the cells. An explicit second order in time, Adams-Bashforth differencing scheme (second order accurate in time) has been used for time advancement of the advection and diffusion terms for each case. Second order central differencing scheme is used for the spatial discretization of both the advection and diffusion term. The boundary conditions used for the present investigation are: (i) at the cross flow inlet, a shear velocity profile of different shear, with its mean being unity is used and the vertical as well as cross stream velocity is considered as zero. (ii) Since our main aim is to study the effect of shear on jet wake and stack wake, we considered a certain portion of the chimney instead of the whole length and applied free slip boundary condition near the bottom of the computational domain except for the zone bounded by the stack. (iii) Free slip boundary condition are imposed on the transverse confining surfaces as well as on the top confining surface. (iv) No-slip boundary conditions are imposed on the walls of the stack. (v) A convective boundary condition as suggested by Orlanski [6] is used as the outlet boundary condition at the exit of the computational domain. For all the computations carried out in the present work the grid size is chosen to be 222 × 114 × 148. The details of the code validation and grid independence test were reported elsewhere [3]. A schematic of the complete computational domain is shown in Fig. 1.

3 Results and Discussion

3.1 Instantaneous Vortical Structures Associated with the Flow Field

To capture the various coherent structures associated with the flow "Q-criterion" technique given by Dubief and Delcayre [7] is used in the present study. Figure 2 presents the vortical structures at BR = 1.0 for K = 0.025, 0.05 and 0.075. The vortical structures near to the stack region and the exit region of the jet are clearly brought into visualization by the iso-surface plots of the vortical structures. The

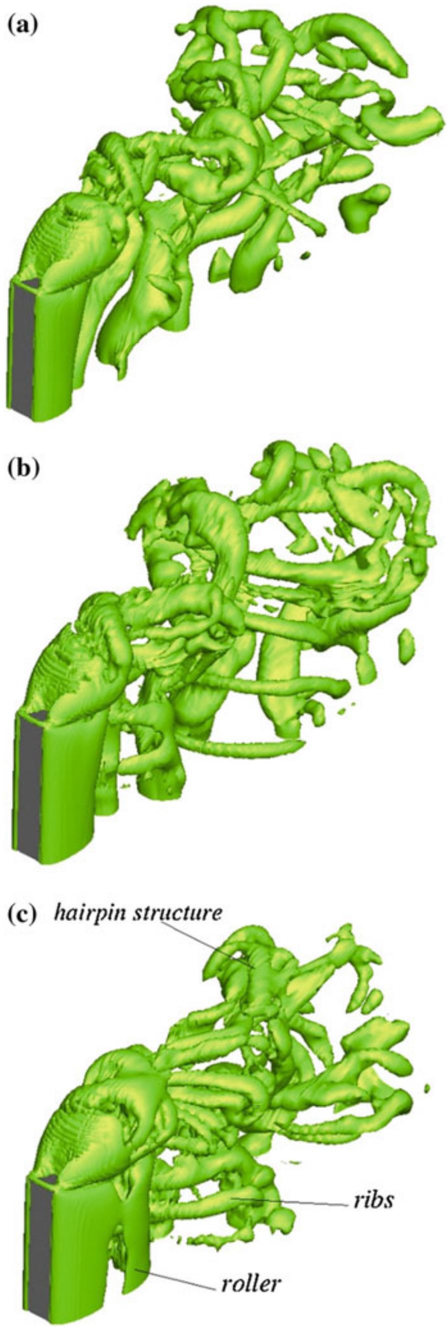

Fig. 2 Instantaneous vortical structures (Q = 0.25) at BR = 1.0 for various shears: **a** K = 0.075, **b** K = 0.05 and **c** K = 0.025

plots of vortical structures clearly shows the presence of hairpin structures, rollers type structures, ribs type structures and stack shear layer. Similar structures associated with EJICF are earlier reported by Arora and Saha [3]. The formation of the roller type structures can be credited to the separating shear layer in the wake of the stack that forms into these structures. However far downstream these rollers are found to be absent, which may be due to the diffusion of vortices with increasing downstream direction. Similar coherent structures (rollers and ribs) are also found for bluff body wakes. A closer look at the structures near the stack reveals the presence of more rib structures in the stack wake for K = 0.025, where as the number of rib like structures is minimum for K = 0.075. The presence of more rib structure in case of K = 0.025 may be due to the interaction between the up-shooting jet and strong downwash present in the flow as compared to the other two cases. The absence of rib structures in case of K = 0.075 can be related to the weaker downwash present in the flow field.

A closer look at the jet exit zone shows the presence of hairpin structures in the wake region of jet and the formation of these hairpin structures are mainly due to the interaction between the cross-flow and the jet coming out of the stack. These hairpin structures are first developed and then shed in the down-stream direction. It is also noted that as the hairpin structures convect down stream they get twisted, mainly because of the shear present in the flow as compared to a flow without shear Arora and Saha [3].

3.2 Instantaneous Flow Field

The instantaneous vorticity contour (ω_z) on the mid-xy plane are shown in Fig. 3. It is evident from Fig. 3 that considerable amount of vorticity is present at the inner walls of the stack. The vorticity generated inside the stack is convected out of the stack by the up shooting jet. Once it comes out of the stack it gets convected down-stream by the action of crossflow. The instantaneous contour of vorticity clearly shows the presence of Kelvin-Helmholtz kind of instabilities. It is also observed from the vorticity contour plot that the immediate near region (along the span of stack) is most affected in case of shear K = 0.025 and least affected incase of shear K = 0.075. This is mainly because of low local blowing ratio in the former case and high local blowing ratio in the latter case.

3.3 Time Averaged Flow Field

The time averaged streamlines on the Z = 0 plane are shown in Fig. 3. To get the average flow field data, all the scalar and vector quantities of the flow field are time averaged over a longer duration of time i.e. around 60 non-dimensional times. The bifurcation lines present in all the three cases are found to be inclined at an angle to